

**KATHMANDU UNIVERSITY**  
**SCHOOL OF ENGINEERING**  
**DEPARTMENT OF MECHANICAL ENGINEERING**

PROJECT REPORT ON



**DESIGN OF HIGH SPECIFIC SPEED FRANCIS TURBINE**  
**FOR LOW HEAD APPLICATIONS**

Dipendra Karki (32159)

Bikalpa Khadka (32160)

Utsarga Lamichhane (32163)

Nikshan Paudel (32168)

JULY 2018

## **ABSTRACT**

In this project, the study has been carried out to investigate the effects of turbine runner parameters on the design. For determining the parameters effects a Francis turbine runner for high specific speed has been designed and the parameter has been studied and numerically analyzed. Design of turbine runner blade is done using Bovet approach. Blade geometry of runner is generated using ANSYS BladeGen software. Numerical analysis has been carried out to determine different parameter's effects on turbine performance. By varying blade outlet angle efficiency and sediment erosion has been compared to design an efficient model of Francis turbine – runner for high specific speed.

## **ACKNOWLEDGEMENT**

We would like to express our sincere gratitude to Dr. Biraj Singh Thapa and Mr. Dadi Ram Dahal for supervising in this work. Without their continuous support and guidance, this project may not have been completed.

We would also like to thank Dr. Hari Prasad Neopane, Mr. Sailesh Chitrakar, Mr. Ram Lama, Mr. Nirmal Acharya and Mr. Saroj Gautam who has provided us with the knowledge and ideas of the project to solve different difficulties faced during the process.

Lastly, we would like to thank everyone at Turbine Testing Lab, KU and all of our colleagues from Department of Mechanical Engineering for their support till the completion of our project.

## TABLE OF CONTENTS

ABSTRACT .....	i
ACKNOWLEDGEMENT .....	ii
LIST OF FIGURES .....	v
LIST OF ABBREVIATIONS .....	vi
LIST OF TABLES .....	vii
LIST OF SYMBOLS.....	viii
CHAPTER 1 INTRODUCTION.....	1
1.1 Context .....	1
1.1 Background.....	2
1.2 Objectives .....	3
1.3 Significance/Scope .....	3
1.4 Limitations.....	3
CHAPTER 2 LITERATURE REVIEW .....	4
2.1 Design approach .....	4
2.2 Francis turbine terminology .....	8
2.3 Computational Fluid Dynamics.....	9
CHAPTER 3 METHODOLOGY .....	13
3.1 Finding Main Dimensions .....	15
3.2 Modeling the Turbine .....	16
3.3 Creating Domain and Mesh.....	16
3.4 Problem Definition in CFD .....	17
3.5 Running Simulation on CFX .....	19
3.6 Observation of Trends of Erosion .....	20
CHAPTER 4 RESULT AND DISCUSSION.....	21

4.1 Efficiency VS Blade Outlet Angle .....	21
4.2 Sediment erosion rate density at various blade outlet angles .....	22
CHAPTER 5 CONCLUSION AND RECOMMENDATION .....	23
5.1 CONCLUSION .....	23
5.2 Recommendations .....	24
APPENDIX .....	25
REFERENCES .....	35

## LIST OF FIGURES

Figure 1 Turbine Selection Chart [2].....	1
Figure 2 Important dimensions of runner channel [2].....	6
Figure 3 Change in blades shape with according to specific rotational speed 'n'[2].....	7
Figure 4 Streamlines on channels' meridional plan [2] .....	7
Figure 5 Bladegen generated meridional profiles .....	8
Figure 6 Flowchart of Francis Turbine Runner Blade Design .....	13
Figure 7 Parameters defined in BladeGen.....	14
Figure 8 Graph produced in BladeGen.....	15
Figure 9 3D geometry of runner .....	16
Figure 10 Mesh obtained from TurboGrid .....	17
Figure 11 Boundary Conditions .....	19
Figure 12 Graph efficiency VS outlet angle .....	21
Figure 13 Sediment erosion rate density comparison.....	22

## **LIST OF ABBREVIATIONS**

ANSYS	Analysis Systems
CFD	Computational Fluid Dynamics
CAD	Computer Aided Design
KW	Kilo Watts
TE	Trailing edge
LE	Leading edge
FEM	Finite Element Method
SST	Shear Stress Transport
PE	Potential Energy
KE	Kinetic Energy
ATM	Automotive Topology Meshing
NPSH	Net Positive Suction Head
NS	Navier- Stoke
R & D	Research and Development

**LIST OF TABLES**

Table 1 Equation of meridional plane parameters [3] ..... 5

Table 2 Design Parameters ..... 15



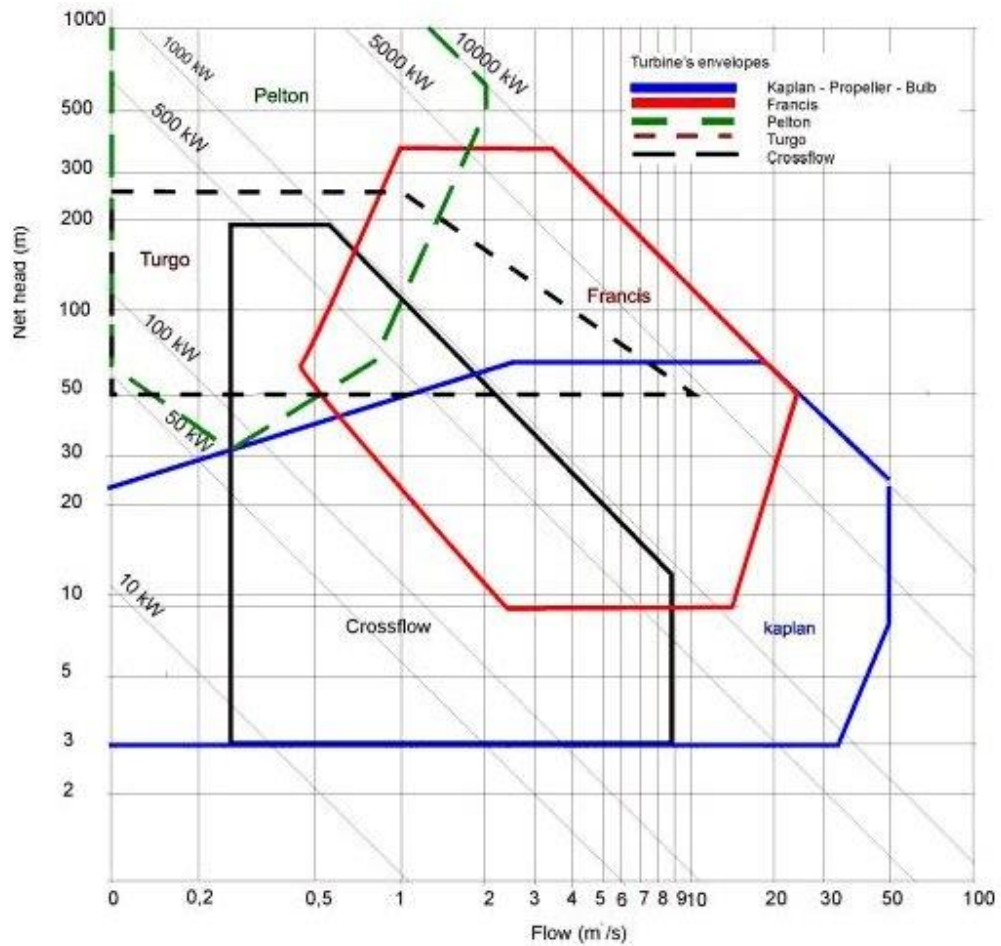
## LIST OF SYMBOLS

$H$	Hydraulic Head	[m]
$Q$	Flow Rate	[m <sup>3</sup> /s]
$N$	Revolution per Minute	[rev/min]
$D$	Runner Diameter	[m]
$C$	Absolute Velocity	[m/s]
$W$	Relative Velocity	[m/s]
$U$	Peripheral Velocity	[m/s]
$n_s$	Specific speed	
$\alpha$	Flow angle(between $C$ and $U$ )	[°]
$\beta$	Flow angle (between $W$ and $U$ )	[°]
$\eta$	Efficiency	
$\rho$	Density	[kg/m <sup>3</sup> ]
$\Gamma$	Circulation	
$b_0$	Guide vane height	[m]
$R_{2e}$	Specific radius	[m]

# CHAPTER 1 INTRODUCTION

## 1.1 Context

Energy demand of the world is increasing day by day and the use of renewable energy sources as hydropower can be a option for the fulfillment of energy demands. As hydroelectricity is the generation of electric power by use of energy stored in water. The main components of a hydropower is the turbine. Generally the basic types of turbines are : pelton, francis, crossflow, kaplan , turgo,etc. The selection of the use of turbines is generally site dependent and according to the turbine selection chart francis turbine has wide range of selection criteria.



**Figure 1** Turbine Selection Chart [2]

The variation range of selection of Francis turbine according to head, flow rate and power output is a better option for easy and efficient type of hydropower construction.

A Francis turbine consists of mainly following components :

**Spiral casing:** The spiral casing around the runner of the turbine is known as the volute casing or scroll case. Throughout its length, it has numerous openings at regular intervals to allow the working fluid to impinge on the blades of the runner. These openings convert the pressure energy of the fluid into momentum energy just before the fluid impinges on the blades. This maintains a constant flow rate despite the fact that numerous openings have been provided for the fluid to enter the blades, as the cross-sectional area of this casing decreases uniformly along the circumference.

**Guide or stay vanes:** The primary function of the guide or stay vanes is to convert the pressure energy of the fluid into the momentum energy. It also serves to direct the flow at design angles to the runner blades.

**Runner blades:** Runner blades are the heart of any turbine. These are the centers where the fluid strikes and the tangential force of the impact causes the shaft of the turbine to rotate, producing torque. Close attention in design of blade angles at inlet and outlet is necessary, as these are major parameters affecting power production.

**Draft tube:** The draft tube is a conduit that connects the runner exit to the tail race where the water is discharged from the turbine. Its primary function is to reduce the velocity of discharged water to minimize the loss of kinetic energy at the outlet. This permits the turbine to be set above the tail water without appreciable drop of available head.

## **1.2 Background**

Since the world is giving priority to renewable energy sources rather than non-renewable fossil fuels. Hydropower being the most important energy source to meet the needs and criteria of energy in the present world .Francis turbine in comparison to others , due to performance, operating expenses and efficiency range allows more versatility in choice of head, flow rate and power. In context of Nepal most of the francis turbines used are for low specific speed and researches related to high specific

speed is not available. This deficiency in efficient design capabilities has led to more research and project to expand design techniques for high specific speed francis turbine.

Performance and efficiency wise it has been seen that francis turbine for low head has more advantages but due to the lack of proper researches and design techniques of high specific speed francis runner it's use has been limited so this project entitled "DESIGN OF HIGH SPECIFIC SPEED FRANCIS TURBINE FOR LOW HEAD APPLICATIONS" can develop an efficient design of high specific speed francis runner to increase it's use in micro hydro.

### **1.3 Objectives**

1. Design low head Francis turbine.
- 2 .Numerical analysis of the runner using CFD technique.
3. Compare efficiency and sediment erosion rate by varying blade outlet angle.

### **1.4 Significance/Scope**

1. Preliminary design will be done using Bovet approach.
2. Profile generation will be done using BladeGen in ANSYS.
3. Design of runner below 50m head.

### **1.5 Limitations**

1. Among different components of Francis turbine runner will be designed.
2. Parametric designs affecting runner design only will be considered.
3. Runner design will be for high specific speed and using Bovet approach.
4. There is no experimental validation of the results obtained from CFD.
5. Economic analysis of cost of efficiency change with increase or decrease in erosion has not been done.

## CHAPTER 2 LITERATURE REVIEW

### 2.1 Design approach

In the design and development of Francis turbine various researches and works have been done to attain the desired efficient model of low specific speed Francis runner. Some of the results and conclusions of the works carried out are as follows:

Lars Froyd (2016) states important features in a low head design:

- 1) The curved shape of the blade leading edge, which results in different inlet diameters at the top and the bottom of the turbine inlet.
- 2) The small difference between inlet and outlet diameter giving a high shroud curvature.
- 3) The straight vertical or sometimes even outward sloping shape of the shroud at the turbine outlet.[1]

Ayancik Fatma(2014) found parametric design based on numerical calculation indicates that hydraulic design parameters causes different effects on the blade shape and the turbine performance. Parametric relations between the beta and theta angle shows that the beta angle distribution has an important role in the design process. [2]

Kazayuki Nakamura and Sadao Kurosawa (2009) optimized runner shape automatically enables on the constraints of the trade off relationship between the runner loss and the draft tube loss. The runner has great improvement on the turbine operating efficiency in the wide operating ranges.[3]

From these literature review it is clear that yvarious works have been done in the field of francis turbine and runner being the most complex and affecting component needs more researches and works based on the high specific speed runner is yet to be done.

### MERIDIONAL PLANE

In the first step of classical design method of a Francis type hydraulic turbine, by the assumption of axial symmetry, the meridional view is obtained. The important dimensions of the runner are calculated by the BOVET. After the dimensions are

determined, we can find the entrance and exit limit curves, that in our case the BOVET method has been used.

Fundamental parameters in turbine design are:

H: Hydraulic head [m]

Q: Turbine flow rate [m<sup>3</sup>/sec]

N: Turbine turns [rpm]

Regarding these parameters, the dimensionless specific speed  $n$ , can be defined as follows:

$$n = \frac{N(P)^{1/2}}{(H)^{5/4}} \quad (1)$$

Specific rotational speed, ‘ $n$ ’ is used to calculate the values of runner channel parameters in the meridional plane. Bovet approach is capable to calculate runner channel parameters in dimensionless form as function of ‘ $n$ ’. Method uses specific radius of ‘ $R_{2e}$ ’ which is the distance between rotational axis of the runner and intersection of shroud and trailing edge of the blade. Specific radius can be calculated from equation (2).

$$R_{2e} = \left( \frac{Q}{\pi} \frac{1}{Nq_{2e}} \right) \quad (2)$$

Where,  $v_{2e}$  represents specific volumetric flow rate. It is taken as 0.270 for this study. To obtain dimensional results, parameters of meridional channel are multiplied with specific radius of the runner. By this way whole variables of the runner channel are reduced to functions of net head, flow rate and rotational speed.

Table 1 Equation of meridional plane parameters [3]

$b_0 = 0.8(2-n)n$	$r_{oi}=y_{mi}=0.7 + 0.16/(n+0.08)$
$l_i=3.2 + 3.2(2-n)n$	$l_e=2.4-1.9(2-n)n$

The correlation between specific rotational speed and meridional channel parameters are not linear as seen in Table 1. So this kind of relation between ‘ $n$ ’ and the parameters causes logarithmic variations and naturally this leads to possibility of designing different types of blades. The BOVET well known relations give the

geometry of hub and shroud curves in the runner blade zone. The form of limit curves is defined by equation:

$$\frac{y}{y_m} = 3.08 \left(1 - \frac{x}{L}\right) \sqrt{\frac{x}{L} \left(1 - \frac{x}{L}\right)} \quad (4)$$

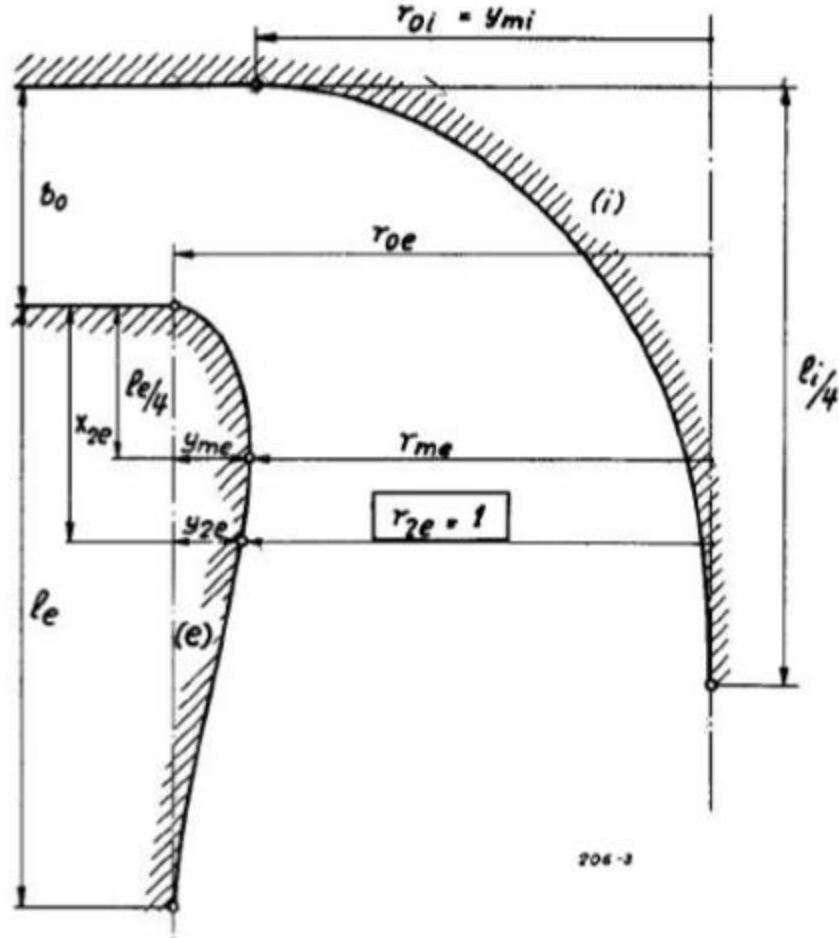


Figure 2 Important dimensions of runner channel [2]

After these necessary calculations are completed, runner channel meridional plane can be created. Then, intersection points of runner blade leading and trailing edges with hub and shroud were determined, respectively. Therefore, runner blade meridional profile was placed on runner channel meridional plane. It is known that the most important design parameter of a runner blade is profile throughout leading to trailing edges. For this purpose, Bovet separates the runner blade with streamlines which have equal flow rate between each other and this is main characteristic of these streamlines. Seven streamlines were used which are named from hub to shroud respectively  $i, i', i'',$

$m, e'', e', e$  where “ $i$ ” spline represents hub curve while “ $e$ ” curve representing shroud (Fig. 4). It guides to design more sensitive geometry while constructing runner in CAD software. Completing construction of meridional profile, it is also need a determination for designate the angle of specific points on the blade. Recent studies are showed that the angle of leading and trailing edges are highly significant when designing high efficient runner. The design approach calculates radiuses and angles at intersection points of streamlines with trailing and leading edges by using velocity triangles.

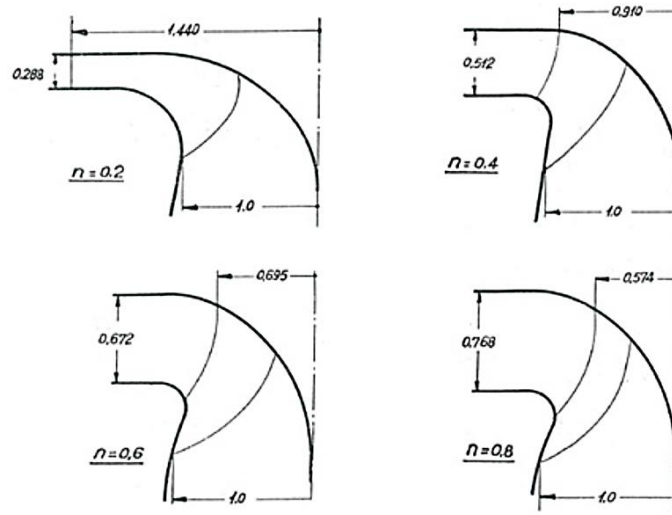


Figure 3 Change in blades shape with according to specific rotational speed ‘ $n$ ’[2]

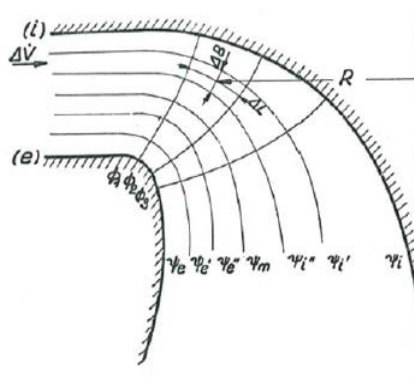


Figure 4 Streamlines on channels' meridional plan [2]



## 2.2 Francis turbine terminology

There are number of terminology related to the Francis turbine.

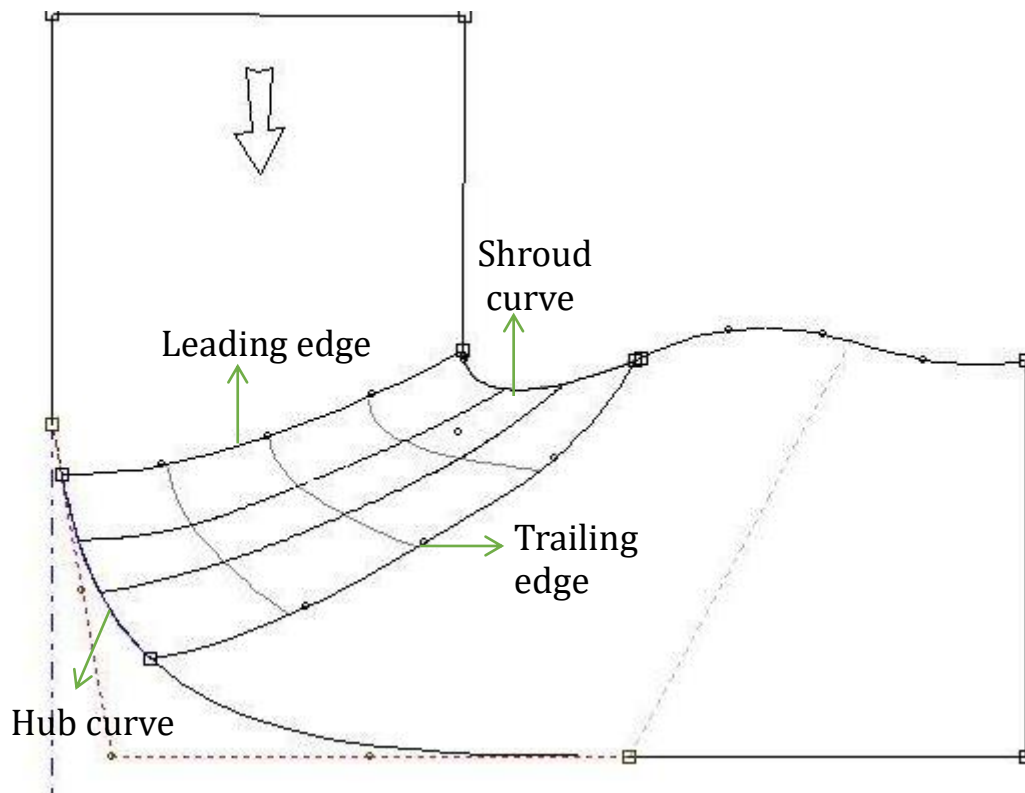


Figure 5 Bladegen generated meridional profiles

**Hub:** It is the side of axis of turbine runner where shaft is attached.

**Shroud:** It is the opposite of the hub as shown in above figure.

**Leading edge:** It is the edge of the Francis runner blade at inlet through which water strike the blade.

**Trailing edge:** It is edge of the Francis runner blade through water discharges to the draft tube.

**Pressure side:** It is the side of the blade at which incoming water strike through inlet.

**Suction side:** It is the side opposite to the pressure side.

**Meridional view:** The view of the blade in the radial and axial coordinate system.

**Blade span:** It is the length of the blade at the inlet through which water impinges at blade.

## **2.3 Computational Fluid Dynamics**

Computational Fluid Dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces studied in science and engineering by iterative process. However, even with simplified equations and high speed supercomputers, only approximate solutions can be achieved in many cases. (Khanna, 2011)

### **2.3.1 CFD Structure**

CFD is structured around the numerical algorithms used to tackle fluid problems. The CFD software includes sophisticated Graphics User Interfaces (GUI) to input problem parameters and to examine the results. The basic components are:

- Pre-processing
- Solver
- Post processing

#### **2.3.1.1 Pre-processing**

It is the first step of analyzing the problem. The activities performed at the pre-processing stage are:

- Definition of the geometry of the region: the computational domain
- Grid generation: the subdivision of the domain into a number of smaller, non-overlapping sub domains.
- Definition of fluid properties
- Specification of appropriate boundary condition at cells, which may coincide with or touch the boundary

#### **2.3.1.2 Solver**

The main part of the CFD structure is the solver. FLUENT, CFX and POLYFLOW are some of the types of solvers. CFX solver is developed independently by ANSYS. It is control-volume based for high accuracy and rely heavily on a pressure based solution technique for broad applicability. The CFX solver uses finite elements (cell

vertex numeric), similar to those used in mechanical analysis, to discretize the domain. The numerical solution of Navier-Stokes (NS) equations in CFD usually implies a discretization method: it means that derivatives in partial differential equations are approximated by algebraic expressions which can be alternatively obtained by means of finite difference or the finite element method. The result is a set of algebraic equations through which mass, momentum, and energy transport are predicted at discrete points in the domain. The governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained.

### **2.3.1.3 Post- processing**

It is the final step in CFD analysis. The result obtained from the solver is interpreted to produce CFD images and animations. (Khanna, 2011)

### **2.3.2 Discretization Methods in CFD**

There are three discretization methods in CFD:

- Finite difference method (FDM)
- Finite volume method (FVM)
- Finite element method (FEM)

#### **2.3.2.1 Finite element method (FEM)**

A finite element method (FEM) discretization is based upon a piecewise representation of the solution in terms of specified basis functions. The discretization involves division of computational domain into smaller domains (finite elements) and the solution in each element is constructed from the basic functions. The actual equations that are solved typically obtained by restating the conservation equation in weak form: the field variables are written in terms of the basic functions; the equation is multiplied by appropriate test functions, and then integrated over an element. Again, a system of equations is obtained (usually for nodal values) that must be solved to obtain a solution.

### 2.3.3 Turbulence Models

The CFD software include the package to model the fluid flow phenomena under the turbulent models. Usually turbulent numerical simulation consists of two main parts, namely: Direct Numerical Simulation (DNS) and Indirect Numerical Simulation (INS). DNS has a precise calculated result, but the whole range of spatial and temporal scales of the turbulence must be resolved which requires a very small time step size. So, this is not suitable for CFD simulations. There are three different types of simulated methods under the indirect Numerical Simulation which are large eddy simulation (LES), Reynolds-averaged Navier Stokes (RANS) and detached eddy simulation (DES). In order to simulate turbulent flows, theoretically, the computational domain should be big enough to contain the biggest eddy. Meanwhile, the mesh should be small enough to find out the smallest eddy. The equation of Reynolds-averaged Navier Stokes (RANS) is defined as:

$$\rho \frac{DU_i}{Dt} = \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \overline{\rho u'_i u'_j} \right] \quad [3]$$

It is the oldest and most common approach to turbulence modeling; the left hand side of the equation describes the change in mean momentum of fluid element and the right side of the equation is the assumption of mean body force and divergence stress.

$\overline{\rho u'_i u'_j}$  is an unknown term and called Reynolds stresses. The RANS equation is not closed due to the presence of stress term, so it requires a turbulence model to produce a closed system of solvable equation. (ANSYS, 2009)

The turbulence model selected is SST which is combine form of k- $\epsilon$  and k- $\omega$  using the advantage of both model.

#### 2.3.3.1 Shear stress Transport (SST) k- $\omega$ Model

The shear- stress transport (SST) k- $\omega$  model was developed by Menter to effectively blend the robust and accurate formulation of the k-  $\omega$  model in the near –wall region with the free-stream independence of the k- $\epsilon$  model in the far field. To achieve this, the k- $\epsilon$  model is converted into a k- $\omega$  formulation. The SST k- $\omega$  model is similar to the standard k- $\omega$  model, but includes the following refinements:

- The standard k- $\omega$  model and the transformed k- $\epsilon$  model are both multiplied by a blending function and both models are added together. The blending function is designed to be one in the near wall region, which activates the standard k- $\omega$  model, and zero away from the surface, which activates the transformed k- $\epsilon$  model.
- The SST model incorporates a damped cross-diffusion derivative term in the  $\omega$  equation.
- The definition of the turbulent viscosity is modified to account for the transport of the turbulent shear stress.
- The modeling constants are different.

#### **2.3.4 Erosion model**

CFX solver has two inbuilt erosion model in it namely Finnie and Tabakoff model.

##### **2.3.4.1 Tabakoff and Grant Erosion Model**

In the erosion model of Tabakoff and Grant, the erosion rate E is determined from the following relations:

$$E = k_1 \cdot f(\gamma) \cdot V_p^2 \cdot \cos^2(\gamma) [1 - R_T^2] + f(V_{PN})$$

Here, E is the dimensionless mass (mass of eroded wall material divided by the mass of the particle),  $V_p$  is the particle impact velocity,  $\gamma$  is the impact angle in radians between the approaching particle track and the wall,  $\gamma_0$  being the angle of maximum erosion,  $k_1$  to  $k_4$ ,  $k_{12}$  and  $\gamma_0$  are model constants and depend on the particle/wall material combination. (ANSYS,2006)

## CHAPTER 3 METHODOLOGY

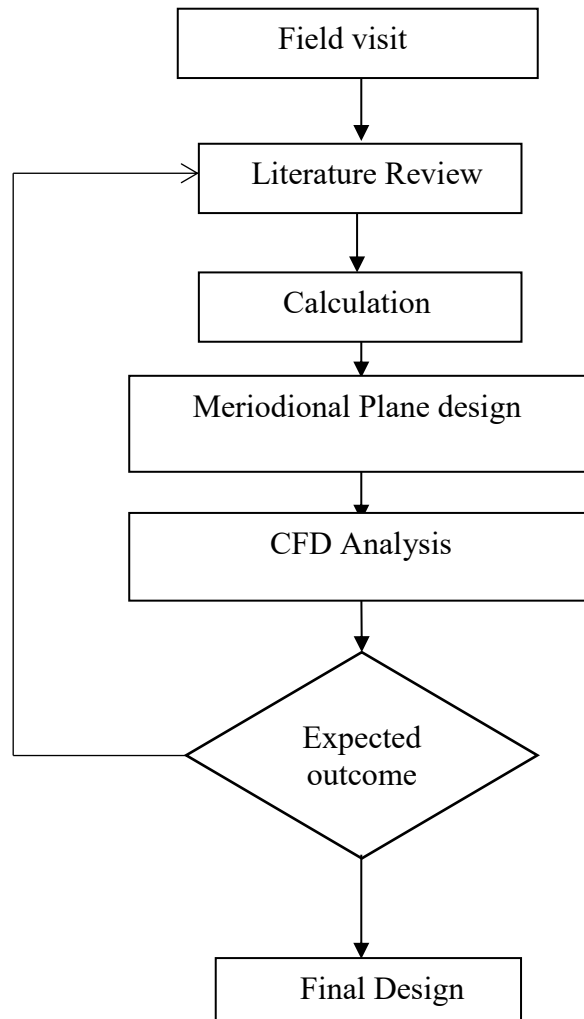


Figure 6 Flowchart of Francis Turbine Runner Blade Design

The various Francis turbine set up at different hydropower stations for high head is being observed during the field visit and the runner design will be generated using Bovet approach. Based on the pre-existing designs and literature review design of the runner for high specific speed has been done in Auto CAD. The designed runner will be analyzed by varying the parametric constraints affecting the design. Based on the results obtained from numerical analysis an efficient design of Francis runner will be proposed.

Techniques used for efficient design of the runner are:

**Bovet approach:** Bovet method uses empirical equations to obtain parameters of Francis type hydro turbine runner [4]. In Bovet method, the dimensionless specific speed value is the main parameter to determine whole turbine dimensions.

**BladeGen:** BladeGen is a feature in ANSYS by which the single blade profile of the runner can be developed. Input parameters of head and flow rate are considered. Prior to the mesh generation for the design and CFD analyses, boundaries of the flow passage should be defined. BladeGen module of ANSYS was selected for the designing because of its comparative advantages. BladeGen provides rapid simulations, modification and optimization when used with ANSYS CFX.

ANSYS BladeGen is a geometry creation tool that is specialized for turbo machinery blades. It is used to generate blades of radial turbine, axial, impeller, radial diffuser desired vane.

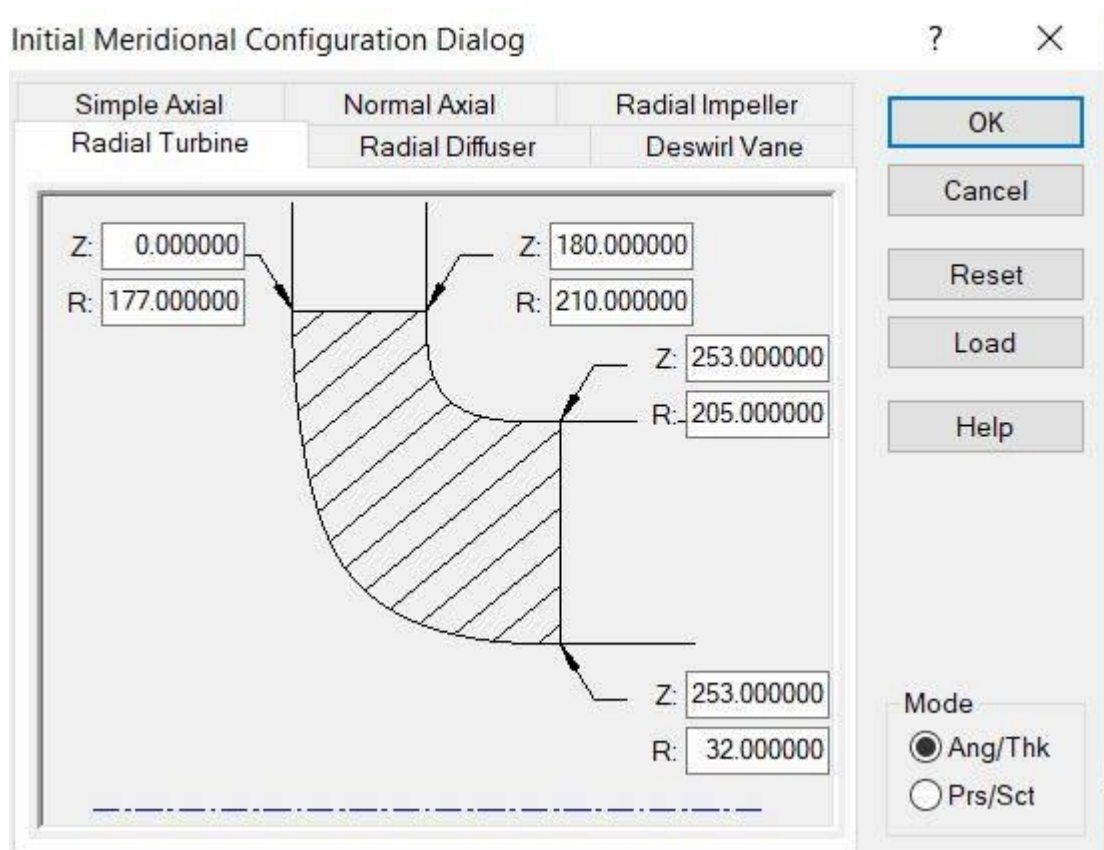


Figure 7 First Bladegen interface to input main dimensions of runner

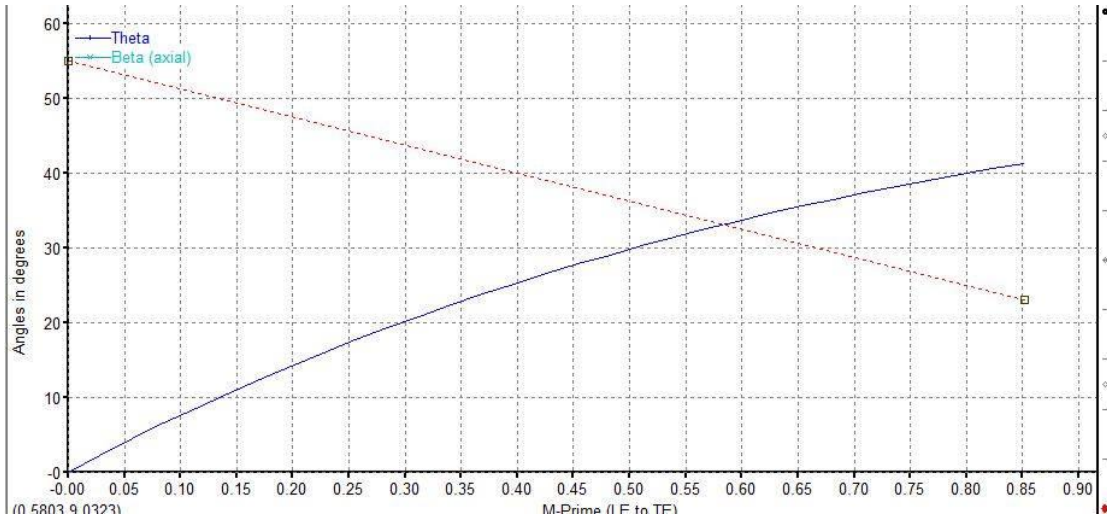


Figure 8 Graph produced in BladeGen

The high specific speed Francis runner has not been developed and the lack of researches has led to this project.

### 3.1 Finding Main Dimensions

Plots for Flow rate and head for different existing Francis turbine were created. And the region with maximum intensity was chosen to set the Q and H of the design.

Table 2 Design Parameters

Specifications		Value
Hydraulic head (H)		30 m
Volumetric flow rate (Q)		1.2 m <sup>3</sup> /s
Rotational speed		750 rpm
Specific speed (n)		186 (from equation 1)
Speed number		0.72 (Low head range)
Specific volumetric flow rate (q <sub>2e</sub> )		0.27
Specific radius (R <sub>2e</sub> )		0.31 (from equation 2)
Meridional channel parameters	b <sub>0</sub>	0.6187 m
	L <sub>i</sub>	5.675 m



	$r_{oi}$	0.964 m
	$L_e$	0.868 m
	$L_i/4$	1.418 m

These dimensions obtained from Bovet method has also been compared with Eulers method and after satisfactory result main dimensions are fixed.

### 3.2 Modeling the Turbine

The modeling of the turbine include the creating 2D view in the BladeGen software which is then converted into the 3D view by applying the beta distribution. This geometry is exported into the Turbogrid which is then meshed using Automotive Topology Meshing (ATM) method.

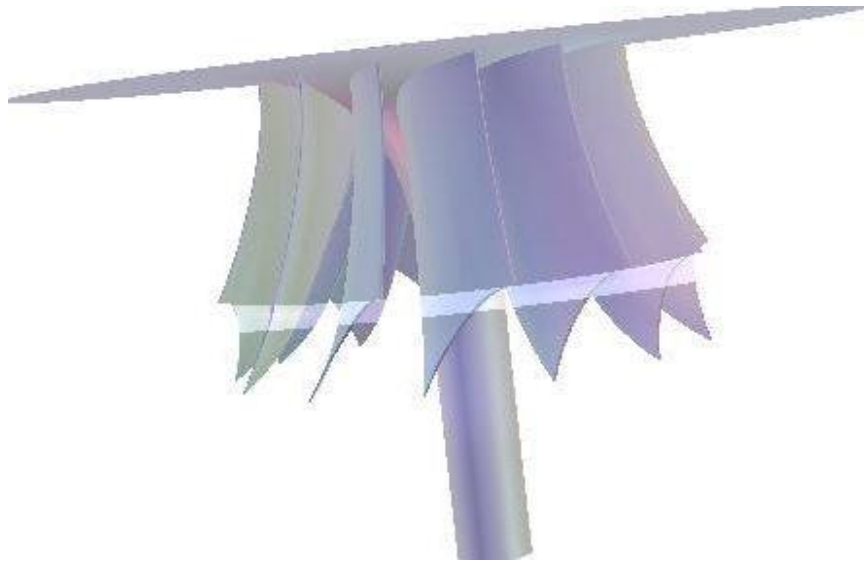


Figure 9 3D geometry of runner

### 3.3 Creating Domain and Mesh

Since very large number of mesh elements are needed to simulate the Francis turbine the domain with only one passage is created that has a blade in the middle. The domain starts from hub wall ends in shroud wall along the span wise direction while its starts from a small distance from the leading edge and ends in a small distance from the trailing edge. The calculations in the domain with whole turbine comprising wicket

gates, stay vanes, spiral casing and draft tube would increase the computational cost as it would take a huge amount of time and require a very powerful computer.

Since the turbine is rotationally periodic it is assumed that the flow in the Francis turbine too is rotationally periodic and simulating in only one passage is enough to visualize the flow in the whole runner.

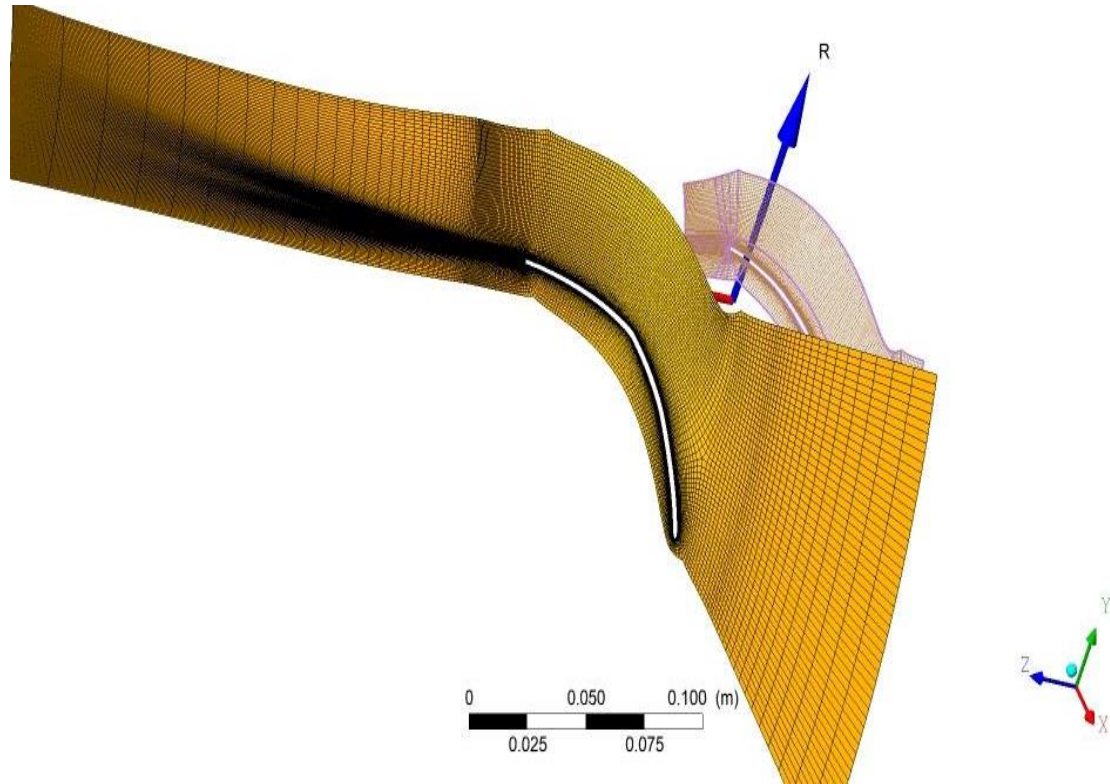


Figure 10 Mesh obtained from TurboGrid

### 3.4 Problem Definition in CFD

Problem definition refers to the setting of the analysis type, adding boundary conditions, choosing turbulence and erosion model in the CFD model. Following were chosen in ANSYS CFX to set up the problem.

### 3.4.1 Choosing Erosion Model

In CFX, there are two namely: finnie and tabakoff model of erosion. The tabakoff model gives the better control on the variable of sediment and control structure. For our purpose the more control available will be better. So, tabakoff model is chosen.

### 3.4.2 Choosing the Turbulence Model

The standard k- $\epsilon$  model is better for flow far from the boundary layer and the standard k- $\omega$  model is near the boundary layer. The Shear Stress Transport (SST) model utilize the positive aspect of the both the models. In Francis turbine runner the flow of water occurs both near the wall and far from the wall in the blade surface and the hub shroud surface. So, the shear stress transport (SST) model is chosen for the simulation.

### 3.4.3. Boundary Conditions

Analysis type: Steady State Analysis

#### Domain:

Fluid & particle definition	Water, Quartz
Reference pressure	1 atm
Erosion modeling	Tabakoff erosion model
Average diameter of quartz	100 micron
Shape factor	Off
Turbulence model	SST model
Analysis type	Steady state
Drag force	Schiller Naumann

**Inlet boundary:**

Flow direction:	Cylindrical components for sand
No. of position	10000
Mass flow rate of quartz	0.07 kg/sec

**Particle control:**

Number of integration steps	10
Maximum particle Integration time step	10s
Tracking time	10 s
Tracking distance	10m
Maximum number integration step	10000

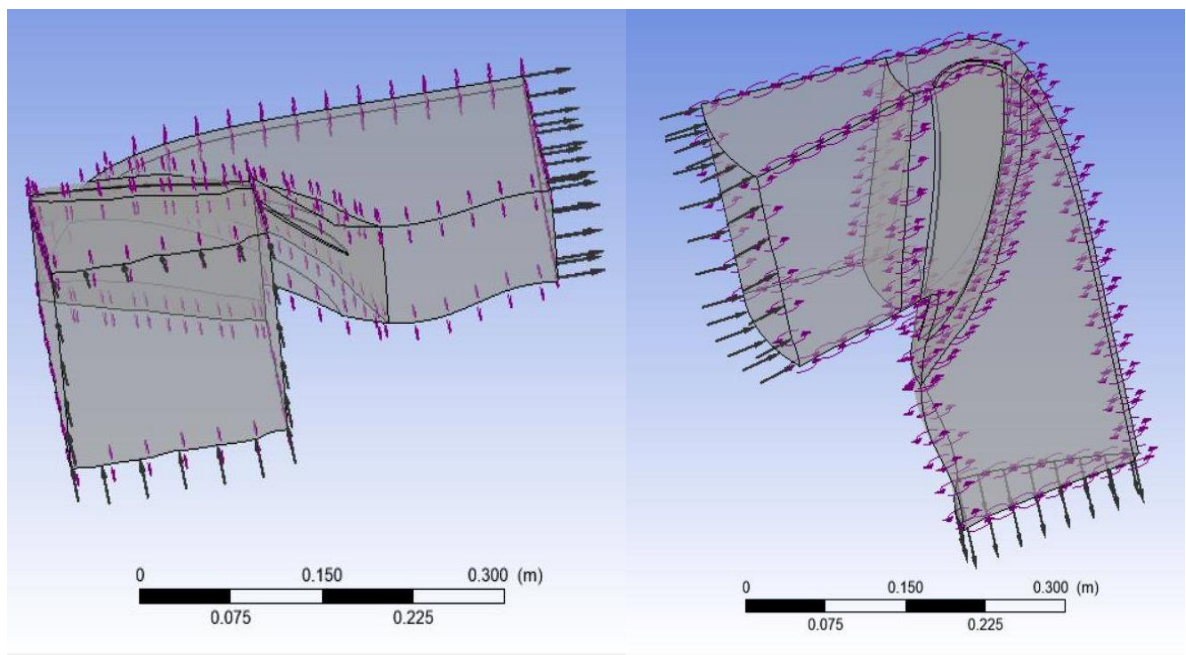


Figure 11 Boundary Conditions

**3.5 Running Simulation on CFX**

The above selected boundary conditions are used in the CFX pre, and .def file is created which is then fed into the solver. The CFX solver solve the

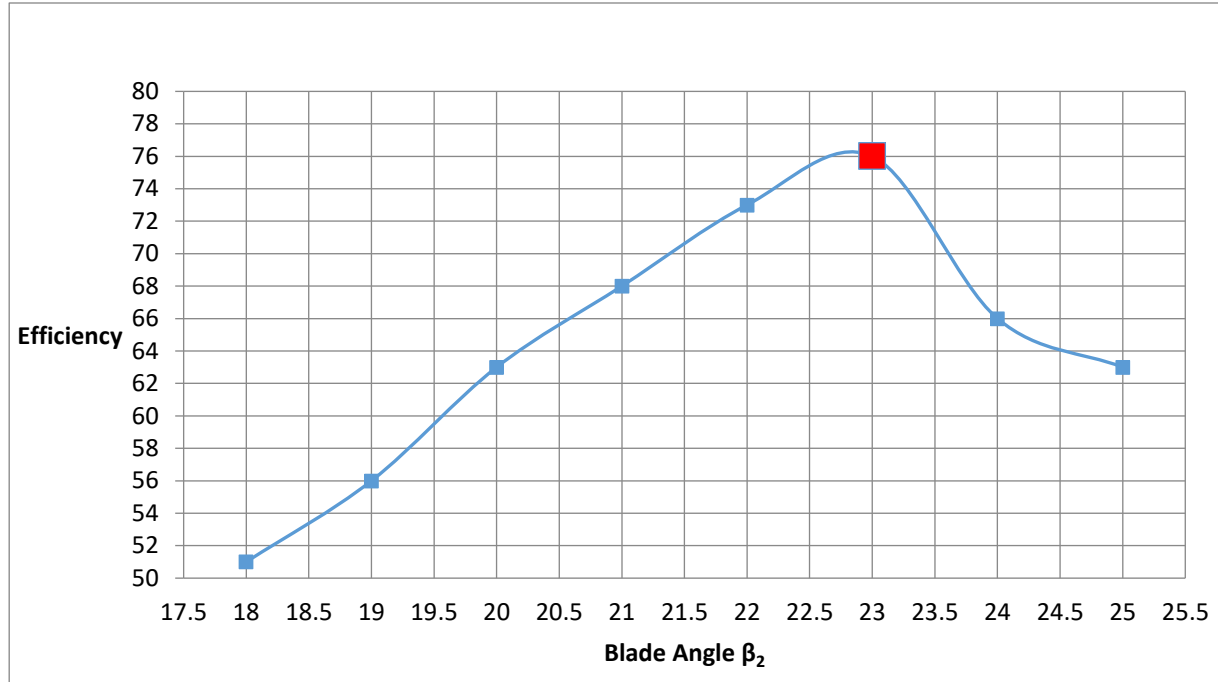
problem and to give the result file. The result thus, obtained is analyze in the CFX post to view the velocity distribution, pressure distribution, erosion, etc. as required.

### **3.6 Observation of Trends of Erosion**

First the variation of erosion with change in blade outlet angle  $\beta_2$  was observed and a value of  $\beta_2$  for minimum erosion was taken for the second kind of model creation.

## CHAPTER 4 RESULT AND DISCUSSION

### 4.1 Efficiency VS Blade Outlet Angle

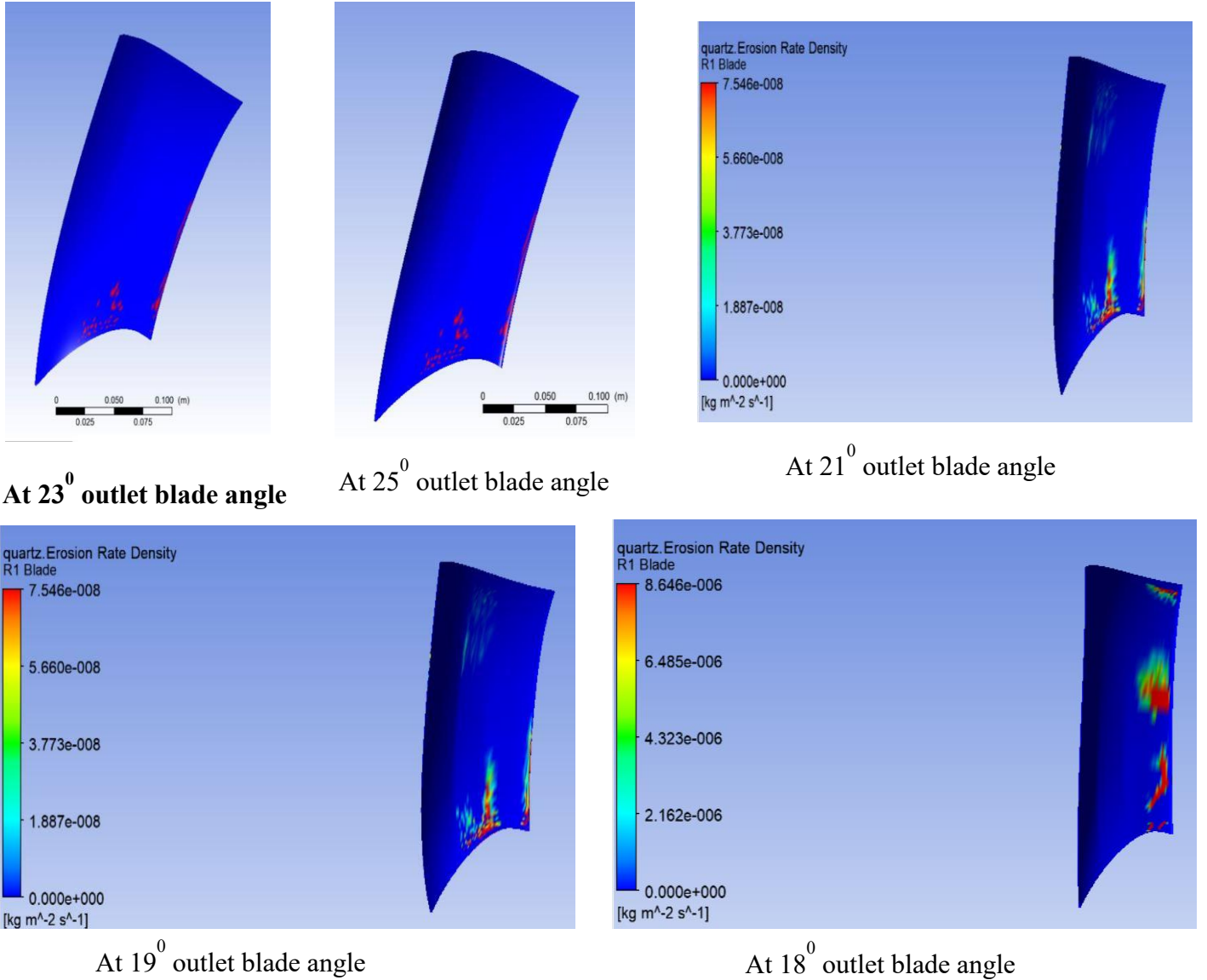


**Figure 12 Graph efficiency VS outlet angle**

By varying the outlet angle from 18° to 25° with 1° span the efficiency of different modeled blade were obtained. The efficiency VS blade outlet angle was plotted and maximum efficiency of the blade was obtained at 23° blade outlet angle and maximum efficiency of 76%.

The trend of efficiency variance with blade angle was: increase in blade angle resulted in the increase in efficiency from 18° to 23° which in the above graph is the line having positive slope. On further increasing the outlet blade angle from 23° the efficiency was found to decrease thus indicating maximum efficiency at blade outlet angle 23°.

## 4.2 Sediment erosion rate density at various blade outlet angles



**Figure 13 Sediment erosion rate density comparison**

The above figure 11 shows the sediment erosion rate density on the suction side of the blades as the blade angle has been increased it has been found that the sediment erosion effect on the blade has decreased. Considering the efficiency and sediment erosion density rate optimum blade design at 23° outlet angle has been chosen.

## CHAPTER 5 CONCLUSION AND RECOMMENDATION

### 5.1 CONCLUSION

With multiple simulations in different models and observing the values of erosion rate density in the blades and efficiency, relation has been figured out between  $\beta_2$  with efficiency and erosion rate density. No clear relation of these parameters can be drawn. However, weak relations can be drawn. Also clear and strong relation of  $\beta_2$  with efficiency can be seen .

The results can be concluded as follows:

- 1) Bovet method is a method that uses empirical relations to determine meridional plane. The results obtained from CFD analysis are shown.
- 2) Higher velocity in the suction sides cause sediment erosion in the suction sides of blades.
- 3) Negative pressure zone in suction side of leading edge and also in trailing edge can cause cavitation. Cavitation formation must be considered
- 4) The erosion was least with no significant decrease in efficiency in the blade of  $\beta_2$  value of  $23^\circ$ .



## **5.2 Recommendations**

- 1) Considerations of draft tube and spiral casing may give more accurate results and improved efficiency.
- 2) Different erosion models can be considered to compare the sediment erosion density rate in the blades.
- 3) Increasing the number of model blades can improve the understanding of the relations of  $\beta_2$  with efficiency and erosion rate density.

## APPENDIX A

### 3D MOELLING OF A FRANCIS RUNNER

Designing of a Francis Runner with Hydrodynamic theory

This theory is to determine the runner dimensions given the flow rate  $Q$  and head  $H$  (Gogstad, 2012).

Steps:

- Choosing the outlet blade angle (between  $13^\circ$  to  $24^\circ$ ) and peripheral velocity of blade inlet  $U_2$  (between 35 to 42 m/s)

$$13^\circ < \beta_2 < 22^\circ \quad \text{lowest value for highest head}$$

$$35 \text{ m/s} < U_2 < 42 \text{ m/s} \quad \text{Highest value for highest head}$$

- Flow at best efficiency point ( $Q^*$ ) =  $Q/1.2$  [ $Q = 1.1$  to  $1.3$  times  $Q^*$ ]

For best efficiency point whirl component of outlet velocity is zero.  $C_{u2} = 0$

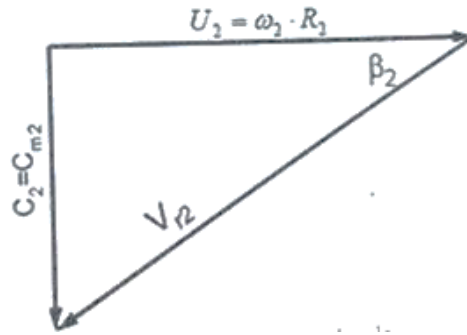


Figure: Outlet velocity triangle

- The meridional velocity at outlet  $C_{m2} = U_2 \tan(\beta_2)$
- The outlet diameter of the runner  $D_2 = \sqrt{\frac{4Q^*}{\pi C_{m2}}}$
- The rpm of the runner  $= \frac{60U_2}{D_2}$

- The rpm of the runner should be synchronized with the frequency of the national grid. For Nepal, it is 50 Hz. The frequency depends upon no. of pole pairs in the generator and the rpm as  $f = \frac{\text{Number of pole pair (Z)} * \text{rpm}}{60}$
- $n(\text{corrected}) = 3000/Z$
- Accordingly the outlet diameter, outlet peripheral velocity, outlet meridional velocity have to be adjusted as:
- $D_2 (\text{corrected}) = \sqrt[3]{\frac{240Q^*}{\pi^2 n \tan \beta_2}}$
- $U_2 = (\pi D_2 n(\text{corrected}))/60$
- $C_{m2} (\text{corrected}) = U_2 * \tan \beta_2$
- The angular velocity of the runner is  $\omega = 2\pi n/60$
- $\omega^* = \omega / \sqrt{2gH}$
- The speed number of the runner is  $\Omega = \omega \sqrt{Q^*}$
- The Net Positive Suction Head Required (NPSH) =  $10 - h_s$  where  $h_s$  is the suction head or submergence height
- The empirical relation for the submergence height is:
- $\text{NPSH} = \frac{(a \cdot C_{m2}^2 + b \cdot U_2^2)}{2g}$  if  $\Omega < 0.55$  then  $a = 1.12$ ,  $b = 0.055$   
If  $\Omega > 0.55$  then  $a = 1.12$ ,  $b = 0.1 \cdot \Omega^*$
- The meridional velocity at the outlet is to be made 10% greater than that of the inlet. Hence we find the meridional velocity at the inlet as
- $C_{m1} = \frac{C_{m2}}{1.1}$  (10 % meridional acceleration)
- The reaction degree of Francis turbine is the ratio of pressure energy to the total energy available to the runner at the inlet. Usually the reaction degree lies in the periphery of 0.5 i.e 50% K.E & 50 % pressure energy. The total energy per mass at the inlet is  $g * h$  and the kinetic energy per mass is  $C_1^2/2$ . Hence the absolute velocity at the inlet is
- $C_1 = \sqrt{(1 - \text{reaction degree}) \times 2gH}$
- The velocity triangle at the outlet of the runner is shown:

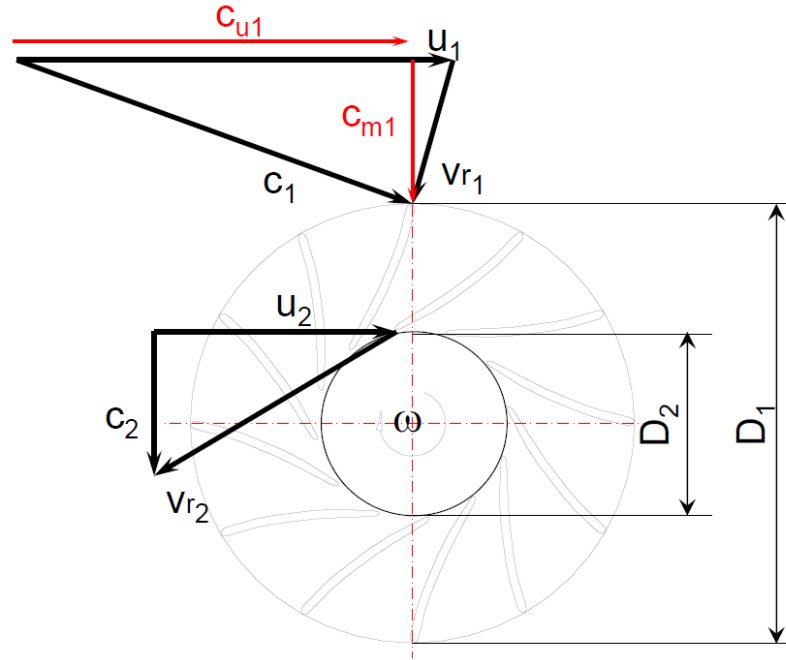


Figure: Velocity triangle in Francis Turbine

The whirl component of the velocity at the inlet is given by:

$$c_{u1} = \sqrt{c_1^2 - c_{m1}^2}$$

- The Euler's equation is given as:
- $\eta = \frac{u_1 c_{u1} - u_2 c_{u2}}{gH}$
- The whirl component at the outlet is zero  $C_{u2} = 0$ . Generally used value of efficiency is 0.96. The peripheral velocity of runner at inlet as:
- $U_1 = \frac{\eta gH}{c_{u1}}$
- The diameter at the inlet is:
- $D_1 = \frac{60U_1}{\pi\eta}$
- The height of the runner at the inlet is:
- $b = \frac{Q^*}{\pi D_1 c_{m1}}$
- The inlet angle of the blade is from the velocity triangle
- $\beta_1 = \tan^{-1} \left( \frac{c_{m1}}{U_1 - c_{u1}} \right)$

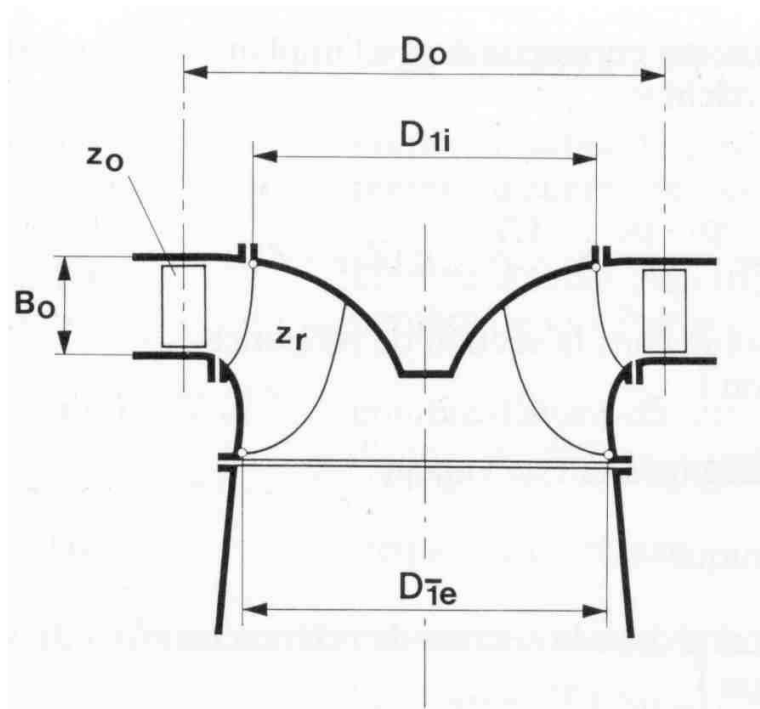


Figure: Turbine main dimensions

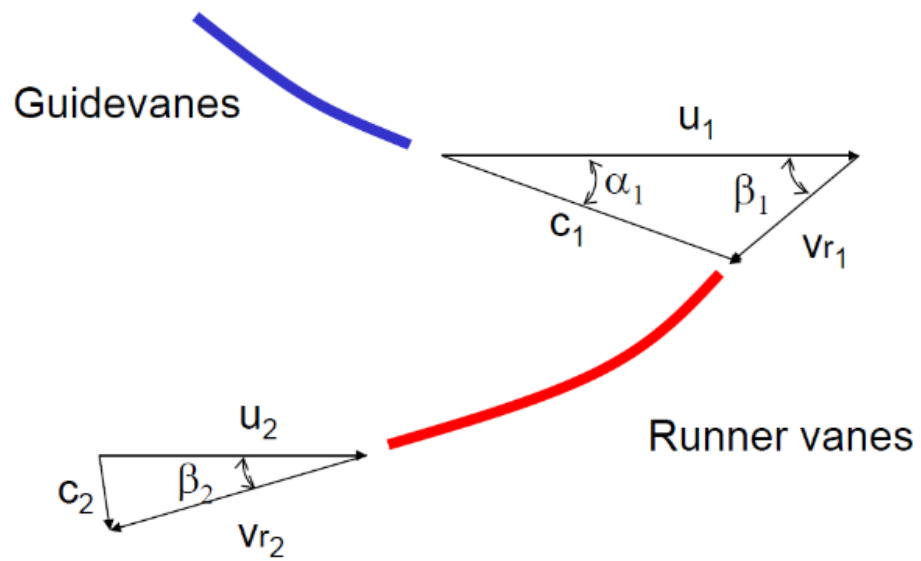
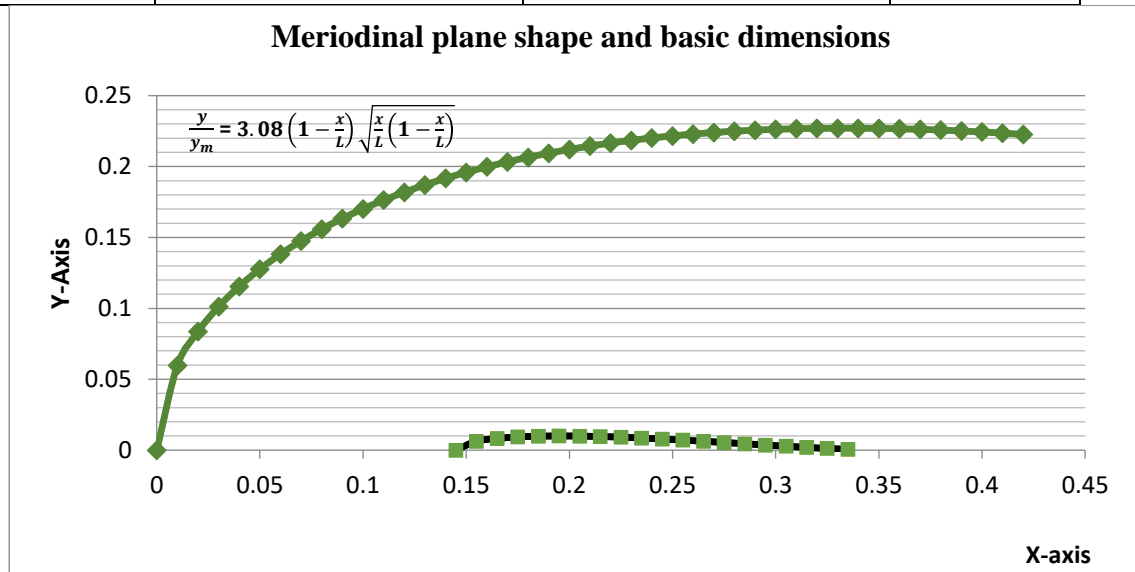


Figure: Velocity triangles

## APPENDIX B

Table: Points plotted for limit curve

x	Y	y	x
0	0	0	0
0.01	0.059788842	0.006314863	0.145
0.02	0.08360022	0.008251452	0.155
0.03	0.101224996	0.009296405	0.165
0.04	0.115545682	0.009826196	0.175
0.05	0.12769285	0.010000818	0.185
0.06	0.138253371	0.009910452	0.195
0.07	0.147580066	0.009614251	0.205
0.08	0.155905584	0.009155029	0.215
0.09	0.163393469	0.008566066	0.225
0.1	0.170164244	0.007874719	0.235
0.11	0.17631	0.007104546	0.245
0.12	0.181903162	0.006276691	0.255
0.13	0.187002038	0.005410876	0.265
0.14	0.191654492	0.004526218	0.275
0.15	0.195900465	0.003642014	0.285
0.16	0.199773761	0.002778666	0.295
0.17	0.203303335	0.001958997	0.305
0.18	0.206514262	0.001210589	0.315
0.19	0.209428454	0.000571042	0.325
0.2	0.212065223	0.00010592	0.335



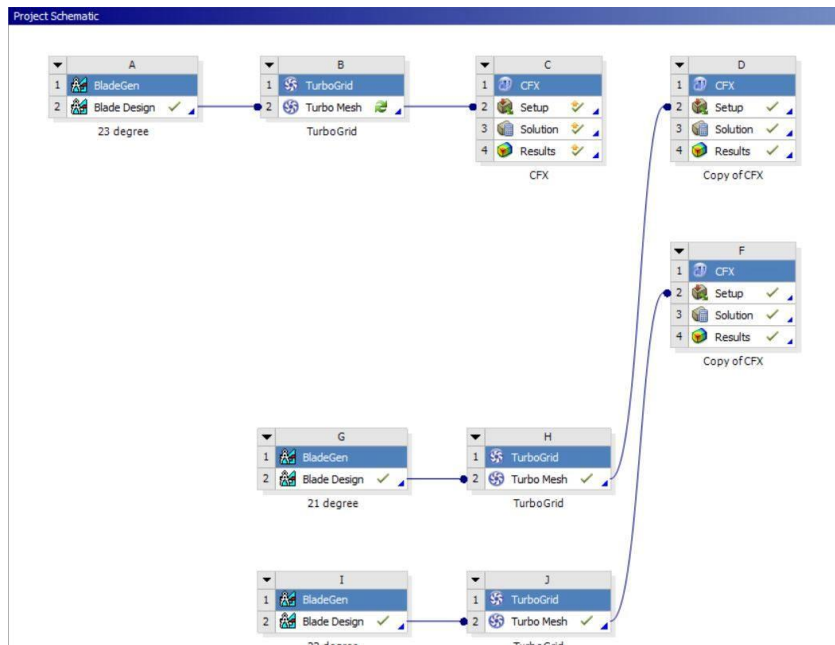


Figure: Stepwise simulation

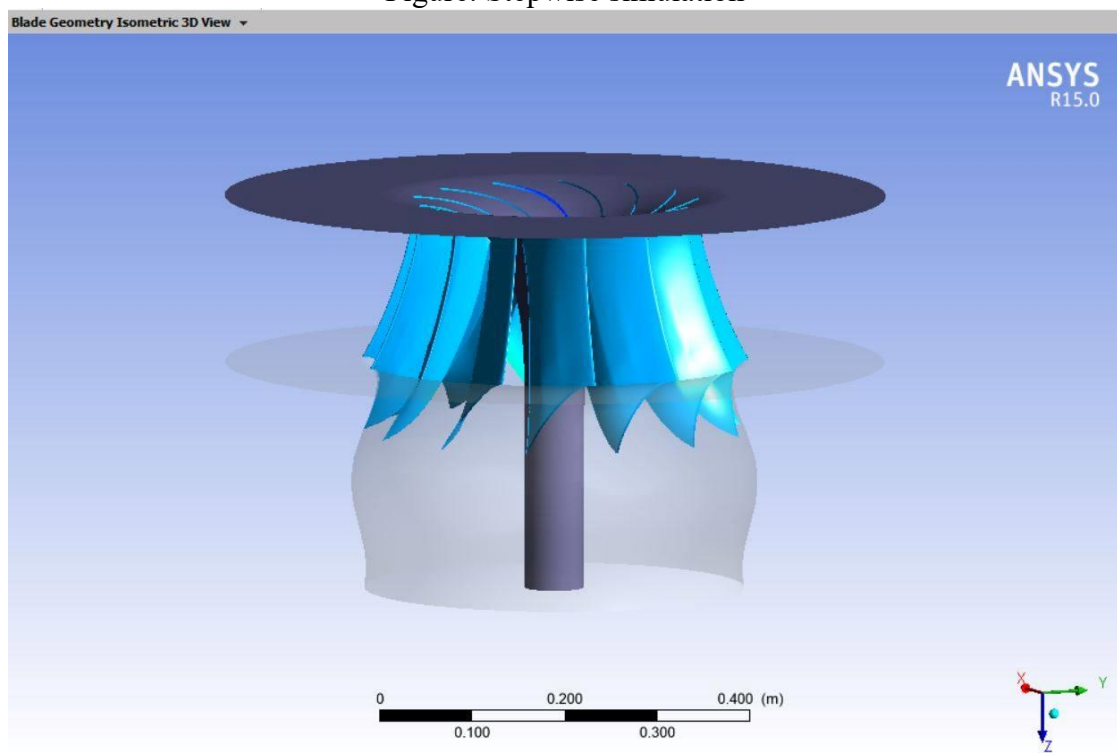


Figure: Blade Geometry isometric 3D view

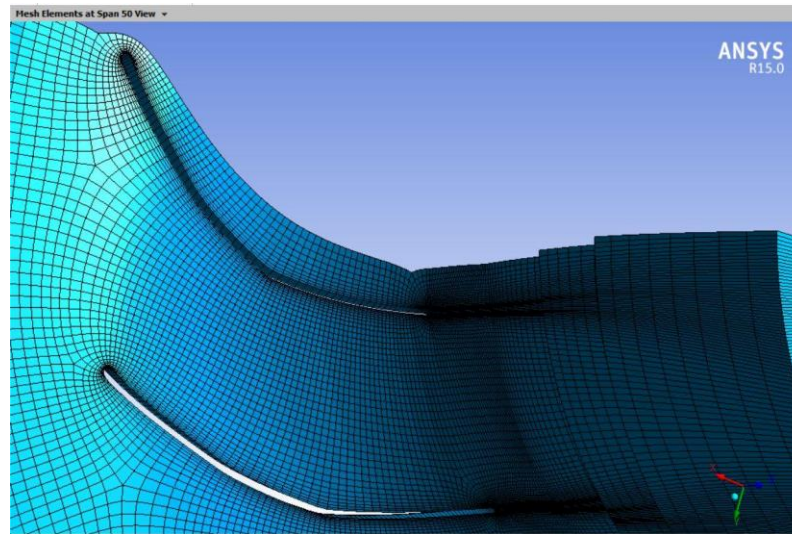


Figure: Mesh Elements in Turbo mesh

Rotation Speed	-78.5398	[radian s <sup>-1</sup> ]
Reference Diameter	0.2712	[m]
Volume Flow Rate	1.2036	[m <sup>3</sup> s <sup>-1</sup> ]
Head (LE-TE)	10.0405	[m]
Head (IN-OUT)	12.3733	[m]
Flow Coefficient	0.7686	
Head Coefficient (IN-OUT)	0.2675	
Shaft Power	98057.5000	[W]
Power Coefficient	0.1385	
Total Efficiency (IN-OUT) %	67.3435	

Figure: Performance results shown by ANSYS with 67% efficiency



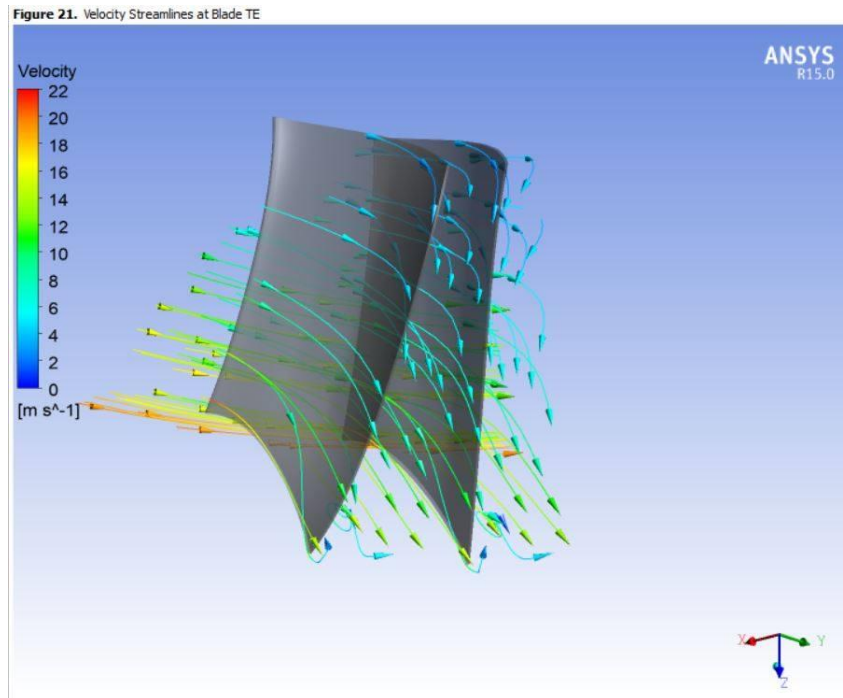


Figure: Velocity Streamlines at Blade Trailing Edge

Rotation Speed	-78.5398	[radian s <sup>-1</sup> ]
Reference Diameter	0.2723	[m]
Volume Flow Rate	0.9204	[m <sup>3</sup> s <sup>-1</sup> ]
Head (LE-TE)	2.7772	[m]
Head (IN-OUT)	3.5408	[m]
Flow Coefficient	0.5805	
Head Coefficient (IN-OUT)	0.0759	
Shaft Power	16940.7000	[W]
Power Coefficient	0.0234	
Total Efficiency (IN-OUT) %	53.1657	

Figure: Performance results shown by ANSYS with 53% efficiency

Rotation Speed	-78.5398	[radian s <sup>-1</sup> ]
Reference Diameter	0.2696	[m]
Volume Flow Rate	1.2036	[m <sup>3</sup> s <sup>-1</sup> ]
Head (LE-TE)	15.2743	[m]
Head (IN-OUT)	18.8010	[m]
Flow Coefficient	0.7819	
Head Coefficient (IN-OUT)	0.4112	
Shaft Power	115024.0000	[W]
Power Coefficient	0.1671	
Total Efficiency (IN-OUT) %	51.9886	

Figure: Performance results shown by ANSYS with 52% efficiency

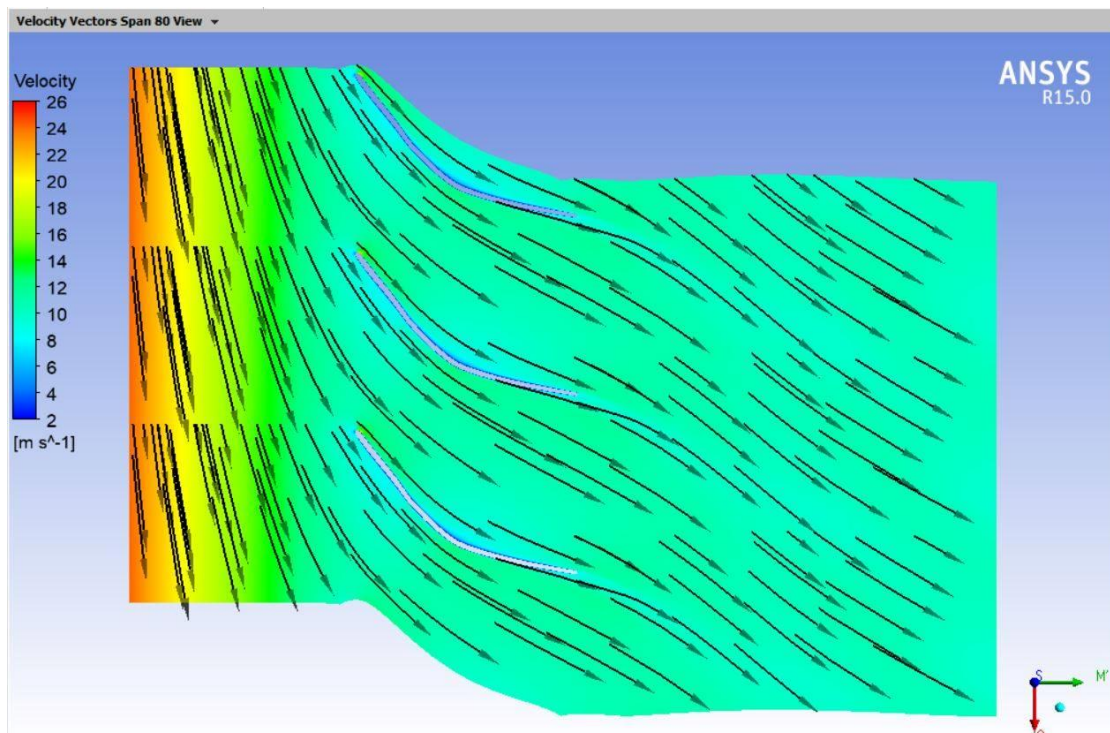


Figure: Velocity vectors span shown in ANSYS

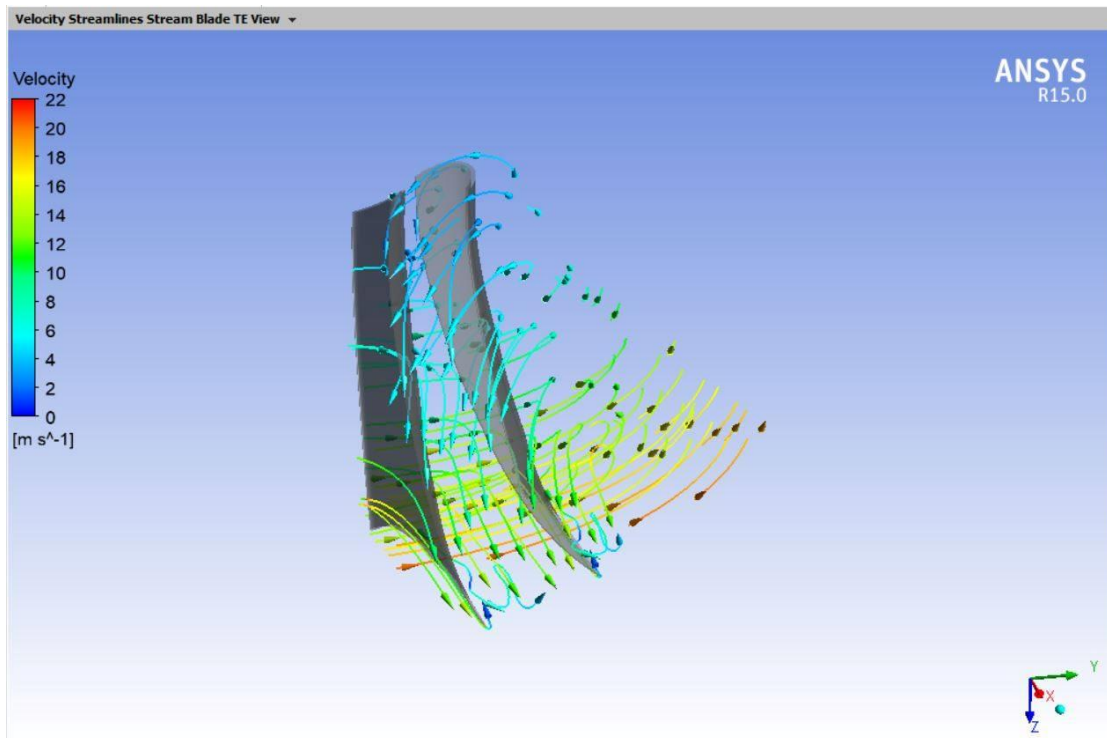


Figure: Velocity Streamlines Stream Blade Trailing Edge

Rotation Speed	-78.5398	[radian s <sup>-1</sup> ]
Reference Diameter	0.2696	[m]
Volume Flow Rate	0.9204	[m <sup>3</sup> s <sup>-1</sup> ]
Head (LE-TE)	2.2775	[m]
Head (IN-OUT)	3.0348	[m]
Flow Coefficient	0.5979	
Head Coefficient (IN-OUT)	0.0664	
Shaft Power	12236.0000	[W]
Power Coefficient	0.0178	
Total Efficiency (IN-OUT) %	44.8037	

Figure: Performance results shown by ANSYS with 45% efficiency

## REFERENCES

- Ayancik Fatma, C. K. (2014). Parametrical and Theoretical Design Of A Francis Turbine Runner With The Help Of Computational Fluid Dynamics. Fluid Mechanics And Thermodynamics, 6.
- Eyup Kocak, S. K. (2016). A numerical case study: Bovet approach to design a Francis runner. ScienceDirect, 10.
- Lars Froyd, K. G. (2016). Development of Design Tool for Low-Head Francis Turbine. Proceedings of the International Symposium on Current Research in Hydraulic Turbines, 6.
- Chitrakar,S., Cervantes,M. & Biraj Singh Thapa , '*Fully coupled FSI analysis of Francis turbines exposed to sediment erosion*', International Journal of Fluid Machinery and Systems (IJFMS), Volume 7, Issue 3, July-September, 2014
- Shrestha, K.P, Chitrakar S., Thapa.B and Dahlhaug,O.G.,'*Performance Comparison of optimized design of Francis Turbines Exposed to Sediment Erosion in various operating conditions*',Journal of Physics: Conference Series, Volume 1042, conference 1
- Marcos S.G. Tsuzuki, S. L. (2015). Development of a Complete Methodology to Reconstruct, Optimize, Analyze and Visulize Francis Turbine Runners. ScienceDirect, 6.
- Neopane, H.P; 2010, *Sediment Erosion in Hydro Turbine*, Trondheim: Norwegian, University of Science and Technology (NTNU)
- Thapa,B;2004, *Sand erosion in hydraulic machinery*,Phd Thesis, Trondheim, Norwegian: Norwegian University of Science and Technology, Faculty of Engineering Science and Technology, Department of energy and process engineering

Thapa, B; Neopane, H.P & Dahlhaug, O.G, 2012, *Particle velocity measurement in swirl flow, laboratory studies*. Kathmandu university Journal of Science, Engineering and Technology, VII(1), p.p, 1-14

